VOLCANIC DEFORMATION MODELLING: NUMERICAL BENCHMARKING WITH COMSOL MULTIPHYSICS

JAMES HICKEY UNIVERSITY OF BRISTOL

The following is a description of the model setups and input/output parameters for benchmarking analytical volcanic deformation models with the commercial Finite Element Analysis package COMSOL Multiphysics, version 4.2a. They should be easily convertible for earlier, or indeed later, versions.

1 The 'Mogi' Model

In the 'Mogi' model, the deformation source is represented by a small point embedded in a homogeneous, isotropic, elastic half-space [*Mogi*, 1958], and the resultant surface displacements are defined by:

$$\begin{pmatrix} U_x \\ U_y \\ U_z \end{pmatrix} = \alpha^3 \Delta P \frac{1-\nu}{G} \begin{pmatrix} \frac{x}{R^3} \\ \frac{y}{R^3} \\ \frac{d}{R^3} \end{pmatrix} \tag{1}$$

where U_x, U_y and U_z are the displacements at the point (x, y, 0) on the free surface, the point source is at (0, 0, -d), α and ΔP are the radius and pressure change of the source respectively, ν is the Poisson ratio, G is the shear modulus (or rigidity) and $R = \sqrt{(x^2 + y^2 + d^2)}$; the distance from the centre of the source to a point on the free surface.

1.1 2D Axisymmetric Setup

1. Open a 2D axisymmetric model in the structural mechanics module using the basic stationary solver.
2. Build the geometry for the model domain to replicate a half-space:
   (a) A square with its top boundary at z = 0 (the free surface), its left boundary at r = 0 (the axis of symmetry), and of sufficient size such that the boundary conditions do not effect the results of the interior.
   (b) A circle with its centre at r = 0, z = -d, and its radius defined. N.B. The radius must be much smaller than the depth (d).
   (c) Select both the square and the circle and press the 'difference' tab so that you are left with a square containing a semi-circle cavity.
   (d) Using 'bézier polygon' (solid type), surround the right and bottom sides of the square with a border as shown in Figure 1.
   (e) Right click on the 'definitions' tab and add an 'infinite element domain'. Select the border area you just created as the domain selection and change the 'geometry' type to 'cylindrical'.
3. Define the material (default is homogeneous, elastic and isotropic):
   (a) Right-click to add a material to the entire model domain and specify the choice of Young's Modulus and Poisson's Ratio. The choice of density will not affect the surface deformation results.

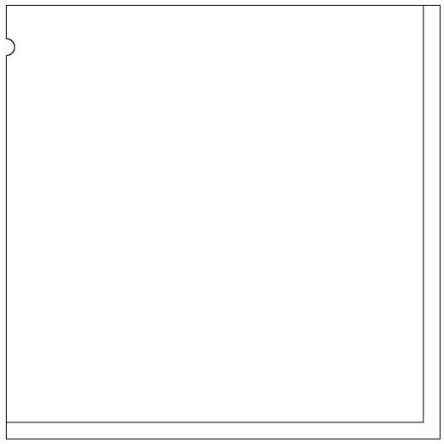

Figure 1: The model geometry used in a 2D axisymmetric representation of the 'Mogi' model. The bordering area to the right and bottom is used to recreate an infinite element domain.

4. Set the model physics:
   (a) Add a 'roller' to the far right boundary of the border to prevent deformation perpendicular to the boundary.
   (b) Add a 'fixed constraint' condition to the bottom boundary to prevent any deformation.
   (c) The top boundary of the square should be 'free' by default.
   (d) Add a 'boundary load' to the two boundaries making the semi-circle. Under 'load type' select 'pressure' and define the necessary pressure value.
5. Define the mesh:
   (a) Add the mesh shape of choice (default is free triangular).
   (b) Set the mesh size to a detail sufficient to resolve necessary displacements. N.B. Pay particular attention to the areas around the source boundaries and the free surface.
6. Run the model.
7. Select the desired data for display:
   (a) For a line plot of deformation at the free surface add a '1D plot group' and then add a 'line graph'. Select the free surface from the boundary selection box and then choose the vertical and/or radial displacement to display.
8. Export data for further analysis:
   (a) For the deformation data along the free surface add a '2D cut line' under 'Data Sets'. Then under 'export', add 'Data' and choose the 2D cut line, then add the data sets you wish to use, e.g. vertical and/or radial deformation.

N.B. In COMSOL v3.5 the boundary load must be applied with the 'tangent and normal' coordinate system.

1.2 3D Setup

1. Open a 3D model in the structural mechanics module using the basic stationary solver.
2. Build the geometry for the model domain to replicate a half-space:

(a) A block with its top boundary at z = 0 (the free surface) and of sufficient size such that the boundary conditions do not effect the results of the interior.

(b) A sphere in the middle of the block, with its centre at x = half the x dimension of the block, y = half the y dimension of the block, z = -d, and its radius defined. N.B. The radius must be much smaller than the depth (d).

(c) Select both the block and the sphere and press the 'difference' tab so that you are left with a block containing a spherical cavity.

(d) Surround the block with separate, further smaller blocks on the bottom and lateral sides to border the initial block, as in Figure 2.

(e) Right click on the 'definitions' tab and add an 'infinite element domain'. Select the border areas you just created as the domain selection and change the 'geometry' type to 'cartesian'.

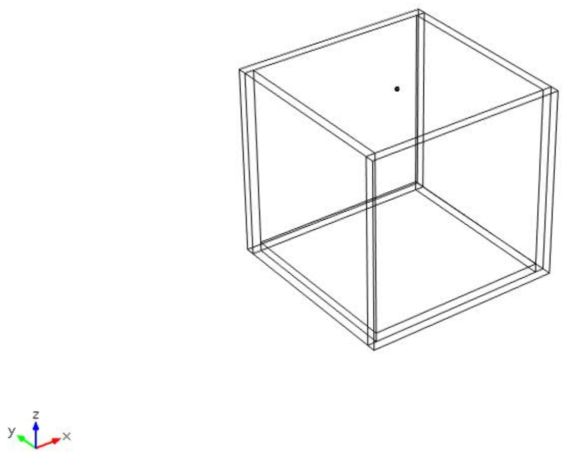

Figure 2: The model geometry used in a 3D axisymmetric representation of the 'Mogi' model. The bordering areas surrounding the initial block are used to recreate an infinite element domain.

3. Define the material (default is homogeneous, elastic and isotropic):

   (a) Right-click to add a material to the entire model domain and specify the choice of Young's Modulus and Poisson's Ratio. The choice of density will not affect the surface deformation results.

4. Set the model physics:

   (a) Add 'rollers' to the lateral boundary surfaces of the borders to prevent deformation perpendicular to the boundary.

   (b) Add a 'fixed constraint' condition to the bottom bordery surface to prevent any deformation.

   (c) The top boundary surface of the block should be 'free' by default.

   (d) Add a 'boundary load' to the eight boundary surfaces making the sphere. Under 'load type' select 'pressure' and define the necessary pressure value.

5. Define the mesh:

   (a) Add the mesh shape of choice (default is free tetrahedral).

   (b) Set the mesh size to a detail sufficient to resolve necessary displacements. N.B. Pay particular attention to the areas around the source boundaries and the free surface.

6. Run the model.

7. Select the desired data for display:

   (a) For a line plot of deformation at the free surface add a '1D plot group' and then add a 'line graph'. Under 'Data Sets' add a '3D cut line' along the necessary transect. Select the '3D cut line' as the data set in the line graph tab and then choose the vertical and/or radial displacement to display.

8. Export data for further analysis:

   (a) Under export, add 'Data' and choose the 3D cut line created for the plot. Then add the data sets you wish to use, e.g. vertical and/or radial deformation.

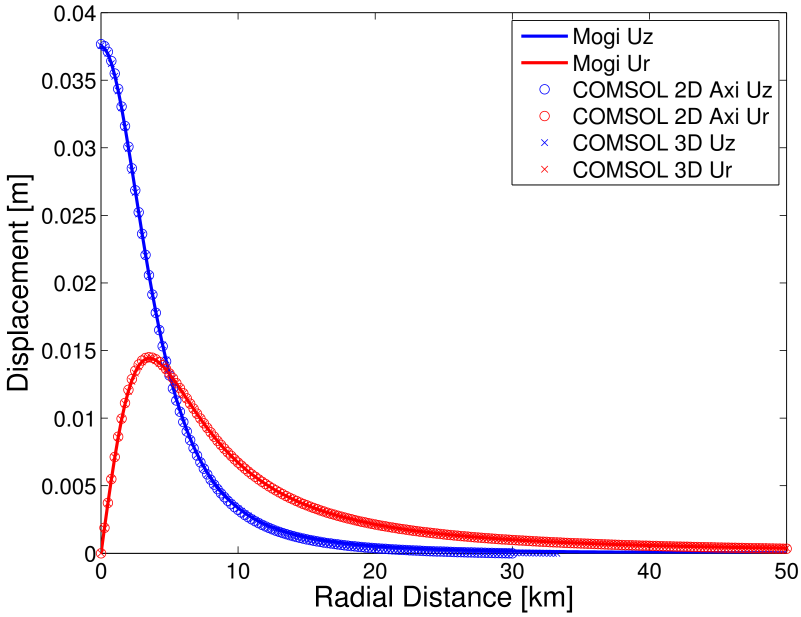

Figure 3: Calibration of the benchmark Finite Element Models with the analytical solution of a point source [*Mogi*, 1958]. The three are nearly perfectly calibrated in both vertical (U_z) and horizontal displacement (U_r).

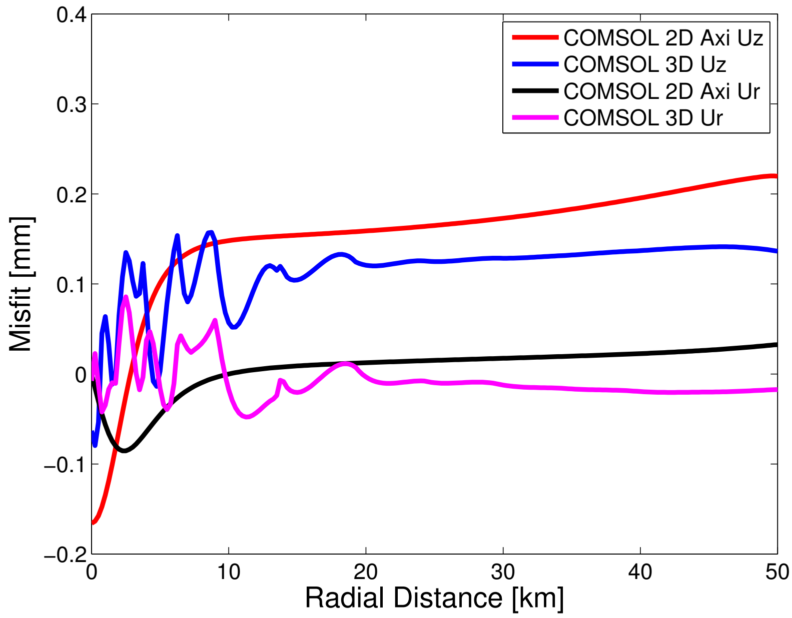

Figure 4: Slight mismatches between the numerical and analytical solutions are thought to be artefacts of the mesh density.

2 Spherical source in a viscoelastic halfspace

To solve for the displacement induced by a spherical source in a viscoelastic half-space, *Del Negro et al.,* [2009] applied the Laplace transform and the Correspondence Principle to the analytical solutions of *Mogi* [1958]. Their model assumes a standard linear solid Maxwell rheology, with one Maxwell viscoelastic branch in parallel with a purely elastic branch (Figure 5).

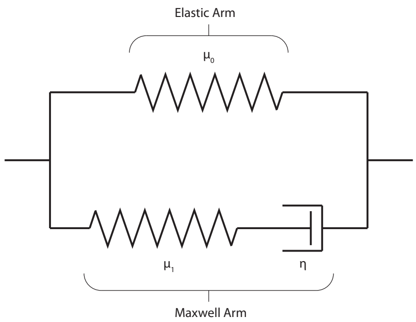

Figure 5: The Standard Linear Solid viscoelastic model.

The resultant surface displacements at time, t, are calculated by:

$$\begin{pmatrix} U_x \\ U_y \\ U_z \end{pmatrix} = \alpha^3 \Delta P \begin{pmatrix} \frac{x}{R^3} \\ \frac{y}{R^3} \\ \frac{d}{R^3} \end{pmatrix} A(t) \tag{2}$$

$$A(t) = \frac{1}{2G_0}\left[\frac{3K+4G_0\mu_0}{\mu_0(3K+G_0\mu_0)} - 3\frac{\eta G_0^2 e^{-(\frac{(G_0\mu_1(3K+G_0\mu_0))}{\eta(3K+G_0)})t}}{\eta(3K+G_0\mu_0)(3K+G_0)}(1-\mu_0) - (\frac{1}{\mu_0}-1)e^{-(\frac{G_0\mu_0\mu_1}{\eta})t}\right] \tag{3}$$

where U_x, U_y and U_z are the displacements at the point (x, y, 0) on the free surface, the source centre is at (0, 0, -d), α and δP are the radius and pressure change of the source respectively, and $R = \sqrt{(x^2+y^2+d^2)}$; the distance from the centre of the source to a point on the free surface. K is the Bulk Modulus and is constant since the viscous part of the deformation is incompressible, and η is the viscosity. G_0 refers to the total shear modulus, which is split equally between the two branches via the fractional shear moduli, μ ($\mu_0 = \mu_1 = 0.5$). The viscoelastic response then depends on the Maxwell relaxation time, $\tau_0 = (\frac{\eta}{G_0\mu_1})$.

2.1 2D Axisymmetric Setup

1. Open a 2D axisymmetric model in the structural mechanics module using the basic time-dependent solver.
2. Build the geometry for the model domain to replicate a half-space:
   (a) A square with its top boundary at z = 0 (the free surface), its left boundary at r = 0 (the axis of symmetry), and of sufficient size such that the boundary conditions do not effect the results of the interior.
   (b) A circle with its centre at r = 0, z = -d, and its radius defined. N.B. The radius must be much smaller than the depth (d).
   (c) Select both the square and the circle and press the ‘difference’ tab so that you are left with a square containing a semi-circle cavity.
   (d) Using ‘bézier polygon’ (solid type), surround the right and bottom sides of the square with a border as shown in Figure 1.

(e) Right click on the ‘definitions’ tab and add an ‘infinite element domain’. Select the border area you just created as the domain selection and change the ‘geometry’ type to ‘cylindrical’.

3. Set the model physics:

   (a) Add a ‘roller’ to the far right boundary of the border to prevent deformation perpendicular to the boundary.
   (b) Add a ‘fixed constraint’ condition to the bottom boundary to prevent any deformation.
   (c) The top boundary of the square should be ‘free’ by default.
   (d) Add a ‘boundary load’ to the two boundaries making the semi-circle. Under ‘load type’ select ‘pressure’ and define the necessary pressure value.
   (e) Add a ‘viscoelastic material’ to the entire model domain and click to specify the Bulk Modulus and Shear Modulus as the material properties. These will be defined later.
   (f) In the ‘solid mechanics’ node choose ‘quasi-static’ for the ‘structural transient behaviour’.

4. Define the material:

   (a) Right-click to add a material to the entire model domain and specify the choice of Bulk Modulus.
   (b) The value for Shear Modulus is split between the two branches of the Maxwell model (one elastic, one viscoelastic. Only enter half of its value here, e.g. if G_0 = 30 GPa only enter 15 GPa.
   (c) The choice of density will not affect the surface deformation results.

5. Set the viscoelastic parameters:

   (a) Under the ‘viscoelastic material model’ node enter the other half of the Shear Modulus in branch 1 of the generalized Maxwell model.
   (b) For the corresponding relaxation time, enter the value (in seconds) equal to the chosen value of viscosity divided by the Shear Modulus of that branch, e.g. if $\eta = 10^{17}$ Pa s and the branch (fractional) Shear Modulus = 15 GPa, then the time value = $6.67\text{x}10^6$s.

6. Define the mesh:

   (a) Add the mesh shape of choice (default is free triangular).
   (b) Set the mesh size to a detail sufficient to resolve necessary displacements. N.B. Pay particular attention to the areas around the source boundaries and the free surface.

7. Set the time dependency:

   (a) In the study step, open the time dependency node and under ‘study settings’ define the time window you wish to use, i.e. start, stop, and step-length.

8. Run the model.

9. Select the desired data for display:

   (a) For a line plot of deformation at the free surface add a ‘1D plot group’ and then add a ‘line graph’. Select the free surface from the boundary selection box and then choose the vertical and/or radial displacement to display.

10. Export data for further analysis:

   (a) For the deformation data along the free surface add a ‘2D cut line’ under ‘Data Sets’. Then under ‘export’, add ‘Data’ and choose the 2D cut line, then add the data sets you wish to use, e.g. vertical and/or radial deformation.

2.2 3D Setup

1. Take the viscoelastic procedure described in section 2.1 and apply it to the 3D ‘Mogi’ model setup in section 1.2.

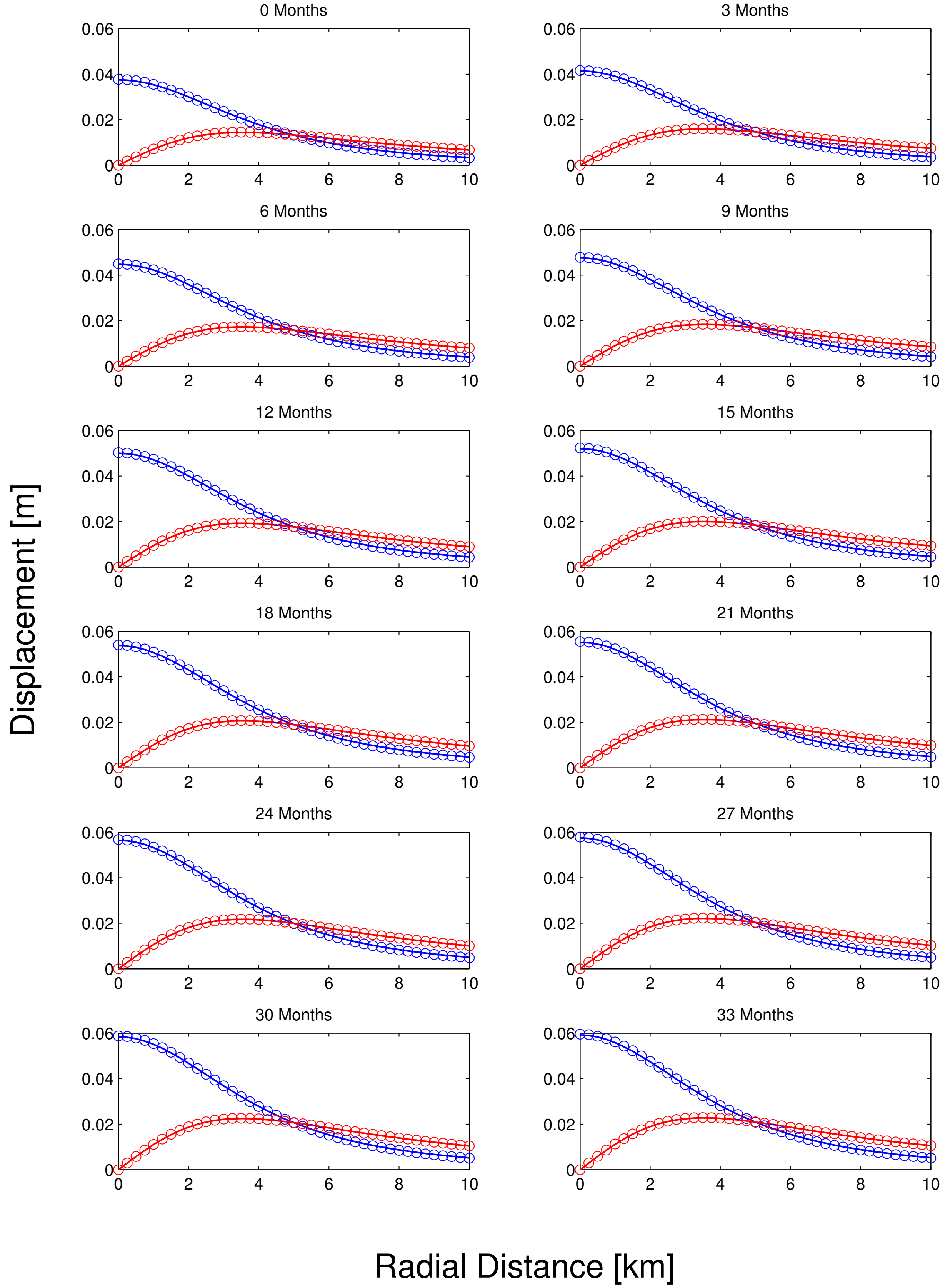

Figure 6: Calibration of the benchmark 2D axisymmetric Finite Element Model with the analytical solution of a spherical source in a viscoelastic half-space [*Del Negro et al.*, 2009]. The solid lines are the analytical solutions for vertical (blue) and radial (red) displacement respectively. The circles are the numerical solutions for the vertical (blue) and radial (red) displacement.

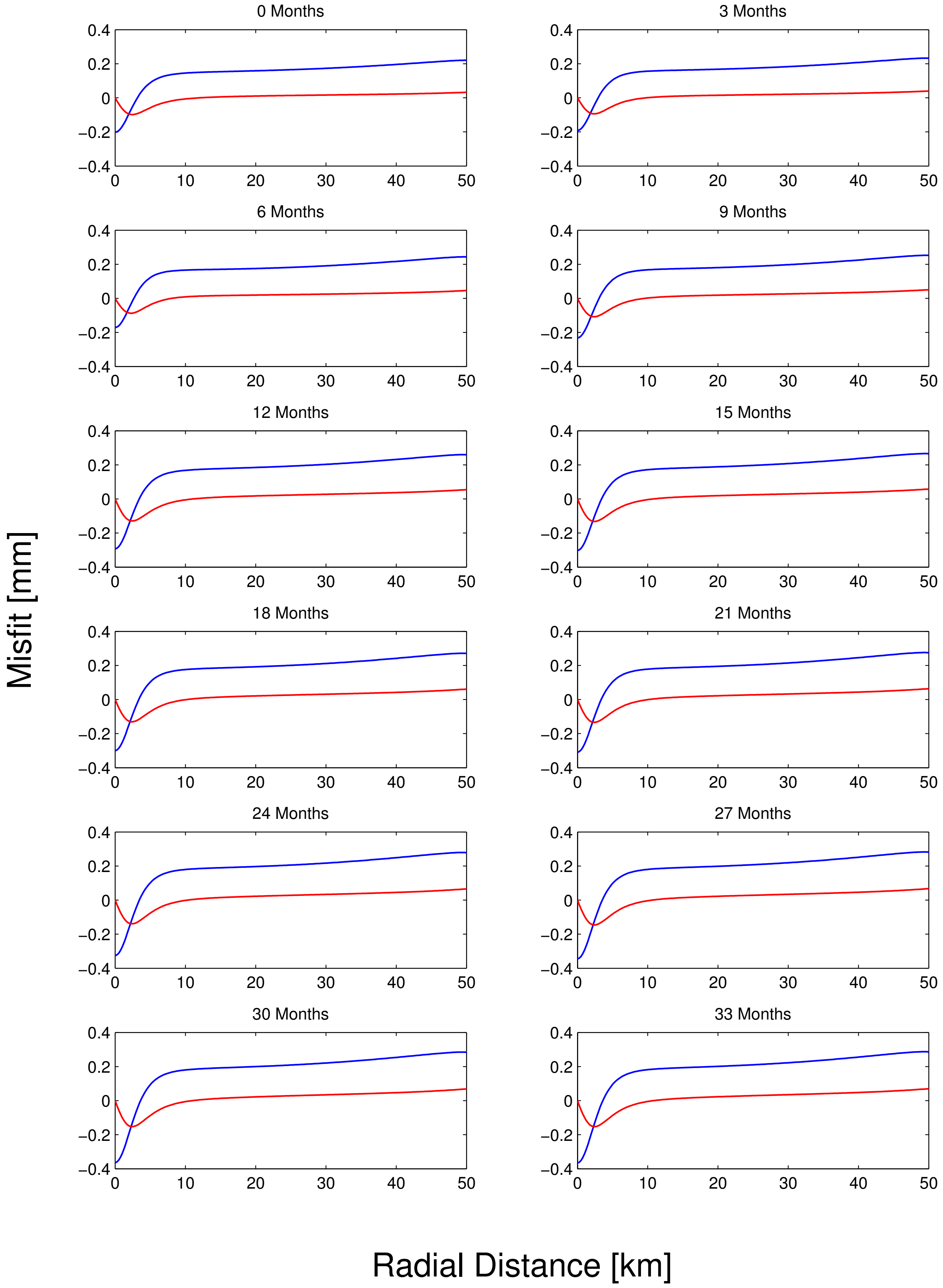

Figure 7: The misfits between the analytical and numerical models for vertical (blue) and radial (red) displacement. Slight mismatches are thought to be artefacts of the mesh density.